

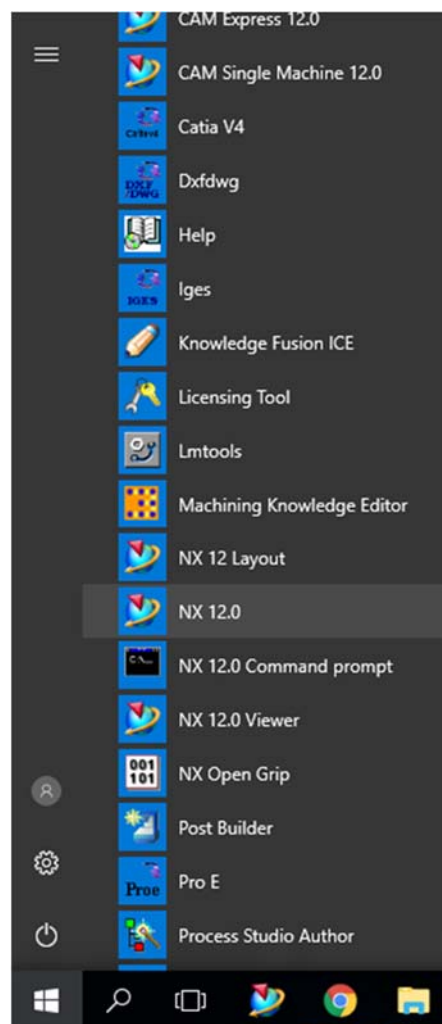
## CHAPTER 2 – GETTING STARTED

Let's begin with starting of an NX 12 session. This chapter will provide the basics required to use any CAD/CAM package. You will learn the preliminary steps to start, to understand and to use the NX 12 package for modeling, drafting, etc. It contains five sub-sections a) Opening an NX 12 session, b) Printing, saving, and closing part files, c) getting acquainted with the NX 12 user interface d) Using layers and e) Understanding important commands and dialogs.

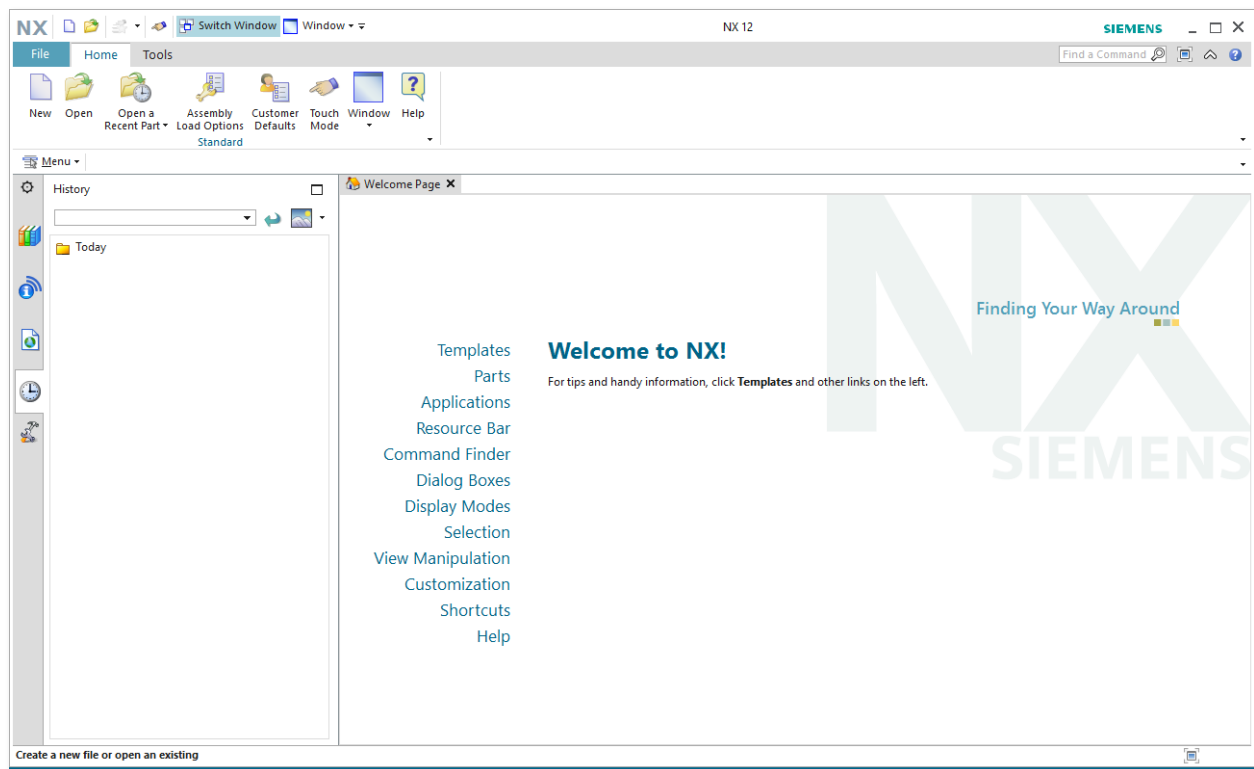
### 2.1 STARTING AN NX 12 SESSION AND OPENING FILES

#### 2.1.1 Start an NX 12 Session

- From the Windows desktop screen, click on **Start** → **Program List** → **Siemens NX 12** → **NX 12**



The main NX 12 Screen will open. This is the Gateway for the NX 12 software. The NX 12 blank screen looks like the figure shown below. There will be several tips displayed on the screen about the special features of the current version. The *Gateway* also has the *Standard Toolbar* that will allow you to create a new file or open an existing file. On the left side of the *Gateway* screen, there is a toolbar called the *Resource Bar* that has menus related to different modules and the ability to define and change the *Role* of the software, view *History* of the software use and so on. This will be explained in detail later in this chapter.



### 2.1.2 Open a New File

Let's begin by learning how to open a new part file in NX 12. To create a new file there are three options.

- Click on the **New** button  on top of the screen

OR

- Go through the **File** drop-down menu at the top-left of the screen and click **New**

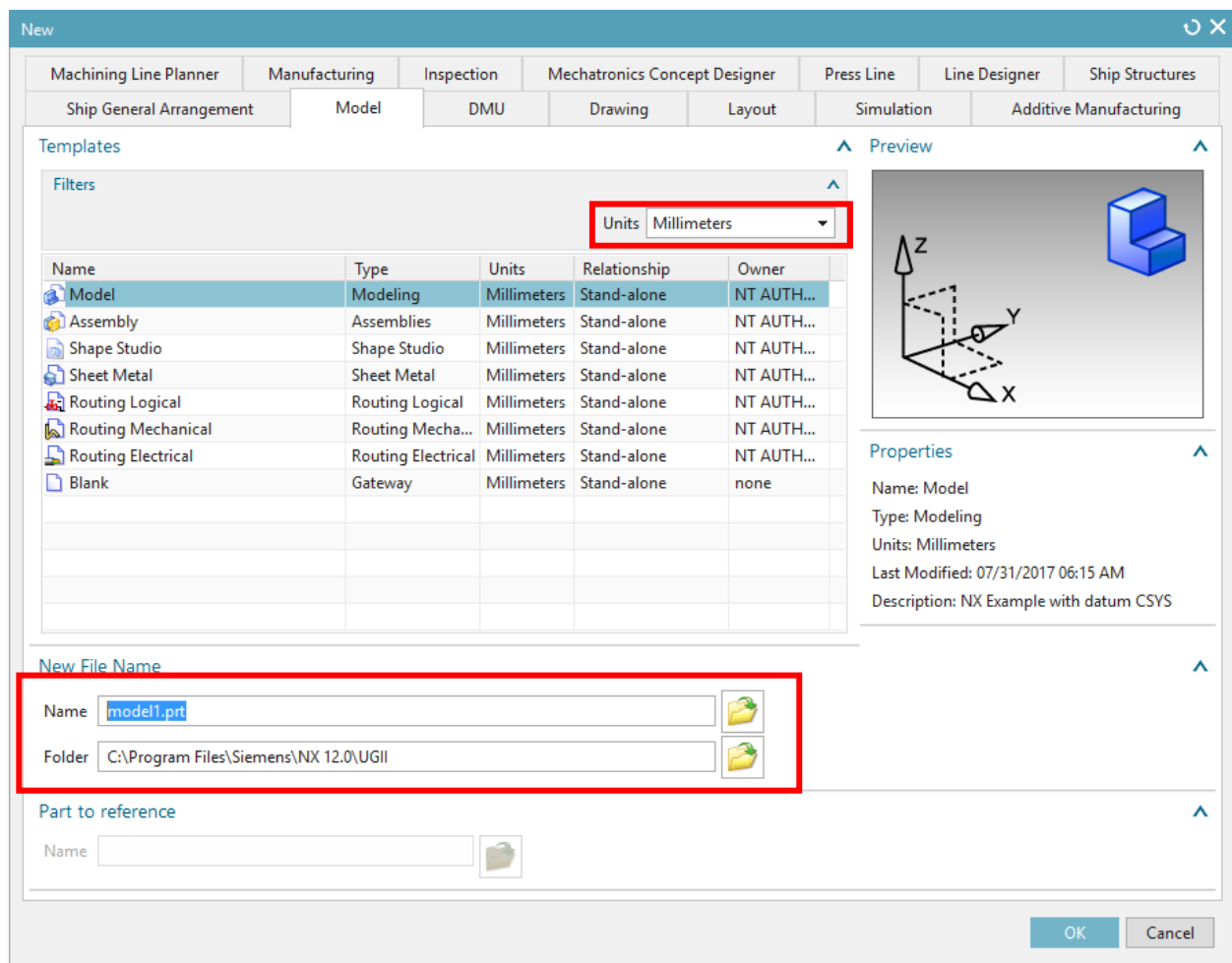
OR

- Press <Ctrl> + N

This will open a new session, asking for the type, name and location of the new file to be created.

There are numerous types of files in NX 12 to select from the *Templates* dialogue box located at the center of the window. The properties of the selected file are displayed below the *Preview* on the right side. Since we want to work in the modeling environment and create new parts, only specify the units (inches or millimeters) of the working environment and the name and location of the file. The default unit is millimeters.

- Enter an appropriate name and location for the file and click **OK**



### 2.1.3 Open a Part File

There are several ways to open an existing file.

- Click on the **Open** or **Open a Recent Part** button on top of the screen

OR

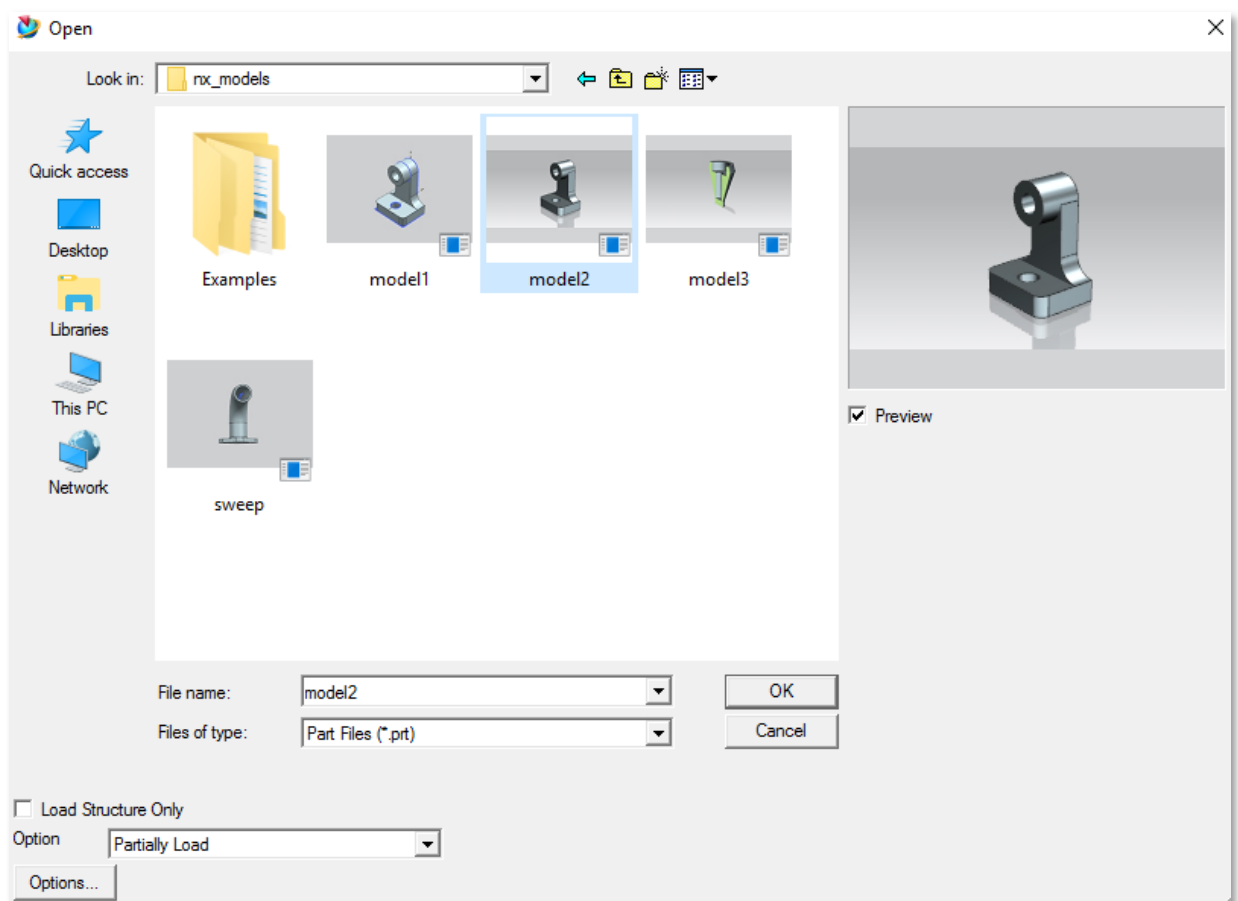
- Go through the **File** drop-down menu at the top-left of the screen and click **Open**

OR

- Press **<Ctrl> + O**

The Open Part File dialog will appear. You can see the preview of the files on the right side of the window. You can disable the *Preview* by un-clicking the box in front of the *Preview* button.

- Click **Cancel** to exit the window



## 2.2 PRINTING, SAVING AND CLOSING FILES

### 2.2.1 Print an NX 12 Image

To print an image from the current display,

- Click **File** → **Print**

The following figure shows the Print dialog box. Here, you can choose the printer to use or specify the number of copies to be printed, size of the paper and so on.

You can also select the scale for all the three dimensions. You can also choose the method of printing, i.e. wireframe, solid model by clicking on the *Output* drop down-menu as shown in the Figure on right side

- Click **Cancel** to exit the window

### 2.2.2 Save Part Files

It is imperative that you save your work frequently. If for some reasons, NX 12 shuts down and the part is not saved, all the work will be lost. To save the part files,

- Click **File** → **Save**

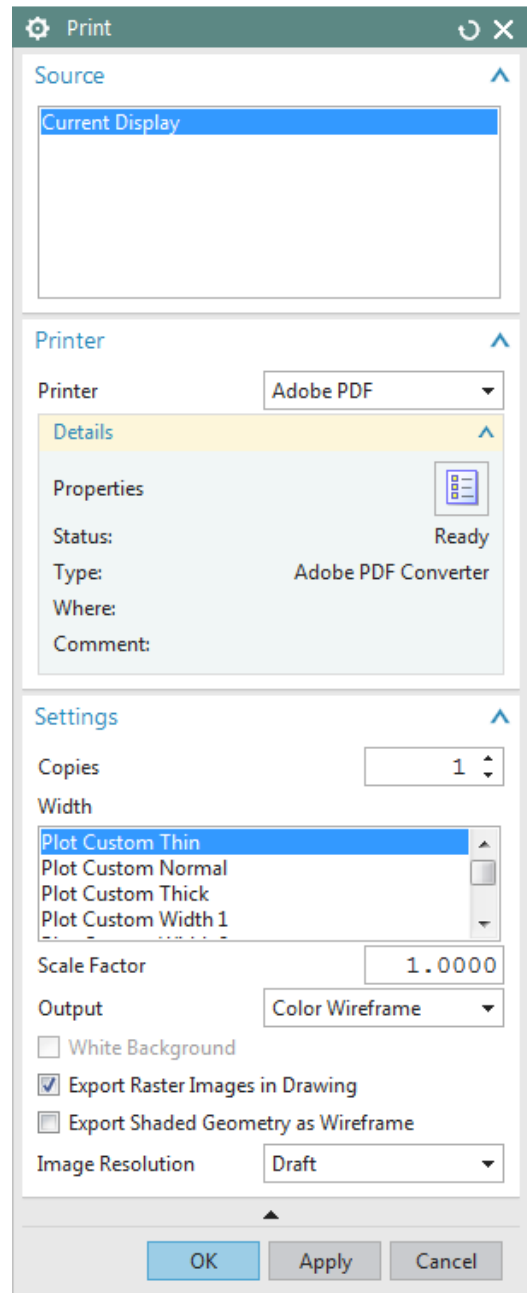
There are five options to save a file:

**Save:** This option will save the part on screen with the same name as given before while creating the part file.

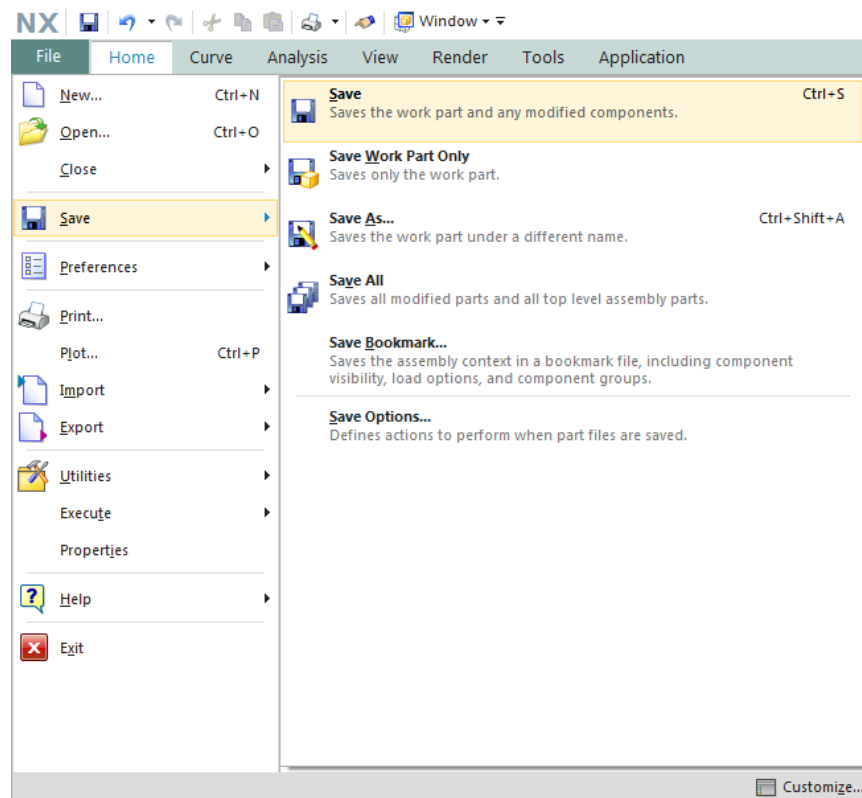
**Save Work Part Only:** This option will only save the active part on the screen.

**Save As:** This option allows you to save the part on screen using a different name and/or type. The default type is *.prt*. However, you can save your file as IGES (*.igs*), STEP 203 (*.stp*), STEP 214 (*.step*), AutoCAD DXF (*.dxf*), AutoCAD DWG (*.dwg*), CATIA Model (*.model*) and CATIA V5 (*.catpart*).

**Save All:** This option will save all the opened part files with their existing names.



**Save Bookmark:** This option will save a screenshot and context of the present model on the screen as a .JPEG file and bookmarks.

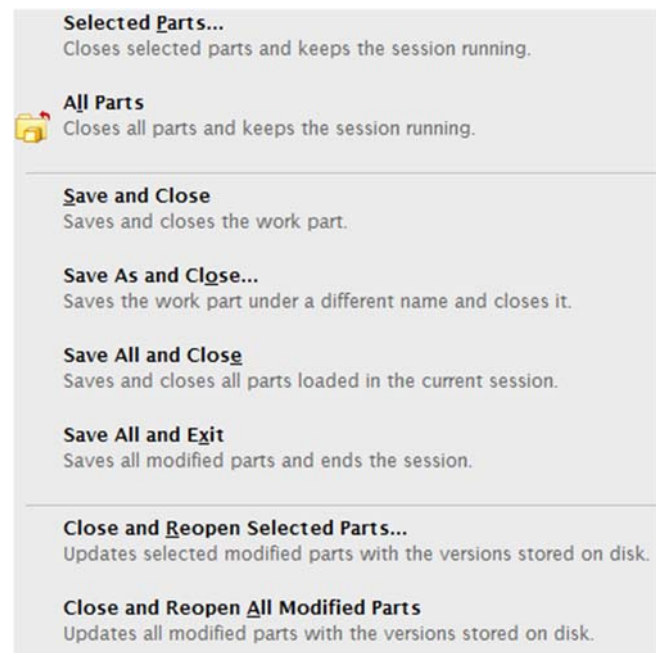


### 2.2.3 Close Part Files

You can choose to close the parts that are visible on screen by

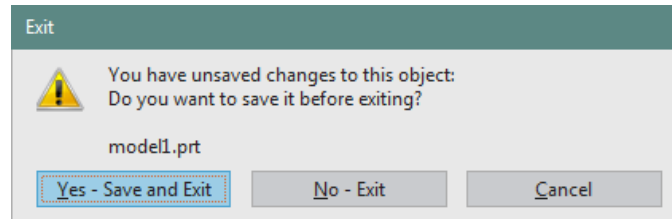
- Click **File → Close**

If you close a file, the file will be cleared from the working memory and any changes that are not saved will be lost. Therefore, remember to select *Save and Close*, *Save As and Close*, *Save All and Close* or *Save All and Exit*. In case of the first three options, the parts that are *selected* or *all parts* will be closed but the NX 12 session keeps on running.



## 2.2.4 Exit an NX 12 Session

- Click **File** → **Exit**



If you have files open and have made changes to them without saving, the message will ask you if you really want to exit.

- Select **No**, save the files and then **Exit**

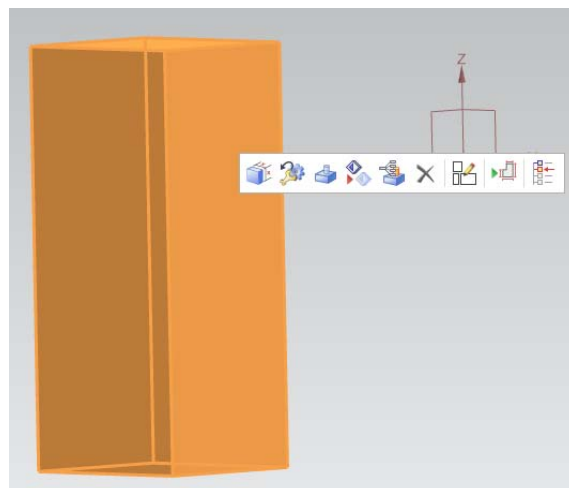
## 2.3 NX 12 INTERFACE

The user interface of NX 12 is made very simple through the use of different icons. Most of the commands can be executed by navigating the mouse around the screen and clicking on the icons. The keyboard entries are mostly limited to entering values and naming files.

### 2.3.1 Mouse Functionality

#### 2.3.1.1 Left Mouse Button (MB1)

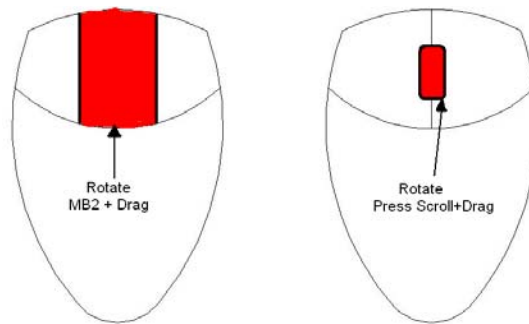
The left mouse button, named Mouse Button 1 (MB1) in NX, is used for *Selection* of icons, menus, and other entities on the graphic screen. Double clicking MB1 on any feature will automatically open the *Edit* Dialog box. Clicking MB1 on an object enables the user to have quick access to several options shown below. These options will be discussed in next chapters.



### 2.3.1.2 Middle Mouse Button (MB2)

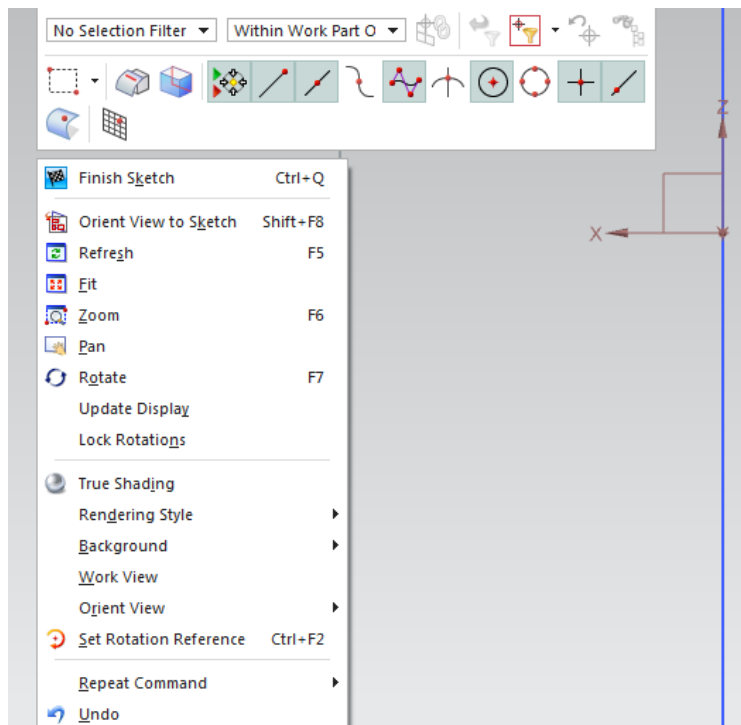
The middle mouse button (MB2) or the scroll button is used to *Rotate* the object by pressing, holding and dragging. The model can also be rotated about a single axis. To rotate about the axis horizontal to the screen, place the mouse pointer near the right edge of the graphic screen and rotate. Similarly, for the vertical axis and the axis perpendicular to the screen, click at the bottom edge and top edge of the screen respectively and rotate. If you keep pressing the MB2 at the same position for a couple of seconds, it will fix the point of rotation (an orange circle symbol appears) and you can drag around the object to view.

If it is a scroll button, the object can be zoomed in and out by scrolling. Clicking the MB2 will also execute the *OK* command if any pop-up window or dialog box is open.



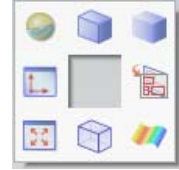
### 2.3.1.3 Right Mouse Button (MB3)

MB3 or Right Mouse Button is used to access the user interface pop-up menus. You can access the subsequent options that pop up depending on the selection mode and *Application*. The figure shown below is in *Sketch Application*. Clicking on MB3 when a feature is selected will give the options related to that feature (Object/Action Menu).





Clicking MB3 and holding the button will display a set of icons around the feature. These icons feature the possible commands that can be applied to the feature.



#### 2.3.1.4 Combination of Buttons

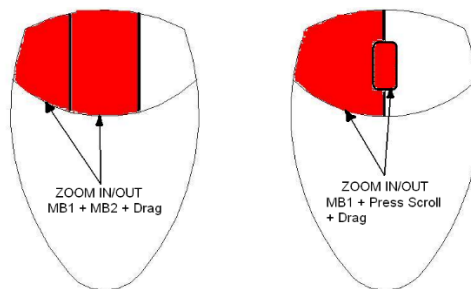
##### Zoom In /Out:

- Press and hold both **MB1** and **MB2** simultaneously and drag

OR

- Press and hold <Ctrl> button on the keyboard and then press and drag the **MB2**

OR

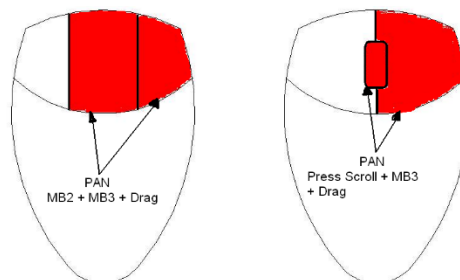


##### Pan:

- Press and hold both the **MB2** and **MB3** simultaneously and drag

OR

- Press and hold <Shift> button on the keyboard and press and drag the **MB2**

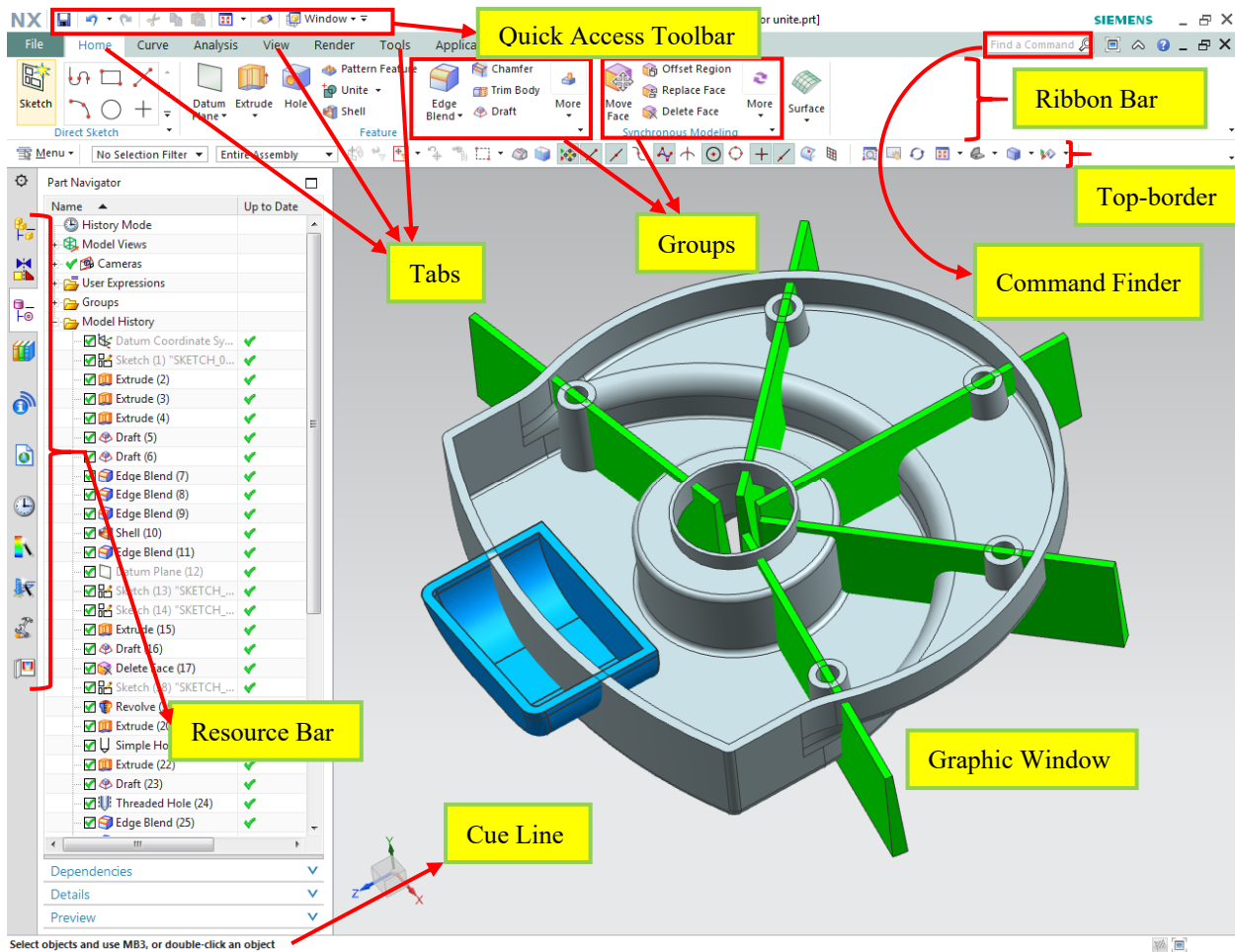


##### Shortcut to menus:

- Press and hold <Ctrl> + <Shift> and **MB1**, **MB2** and **MB3** to see shortcuts to **Feature**, **Direct Sketch**, and **Synchronous Modeling** groups, respectively

## 2.3.2 NX 12 Gateway

The following figure shows the typical layout of the NX 12 window when a file is opened. This is the Gateway of NX 12 from where you can select any module to work on such as modeling, manufacturing, etc. It has to be noted that these toolbars may not be exactly on the same position of the screen as shown below. The toolbars can be placed at any location or position on the screen. Look out for the same set of icons.



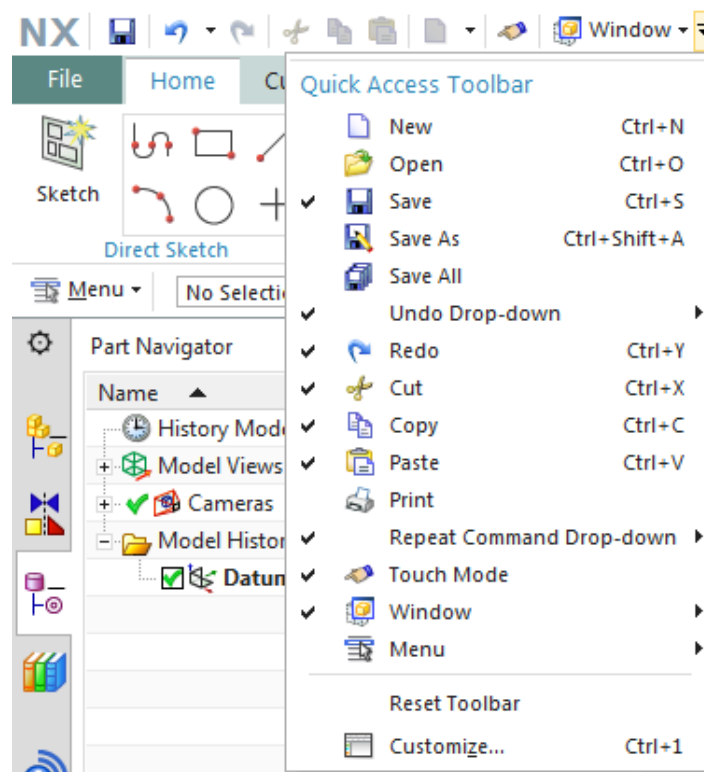
### 2.3.2.1 Ribbon Bar

The ribbon bar interface gives the user the ability to access the different commands easily without reducing the graphics window area. Commands are organized in ribbon bars under different tabs and groups for easy recognition and accessibility.

For example in the ribbon bar shown in the figure above, we have home, curve, etc. tabs. In the home tab, we have direct sketch, feature, synchronous modeling and surface groups. And in each group, we have a set of featured commands.

### 2.3.2.2 Quick Access Toolbar

The quick access toolbar has most commonly used buttons (save, undo, redo, cut, copy, paste and recent commands) to expedite the modeling process. You may easily customize these buttons as shown in the figure below.



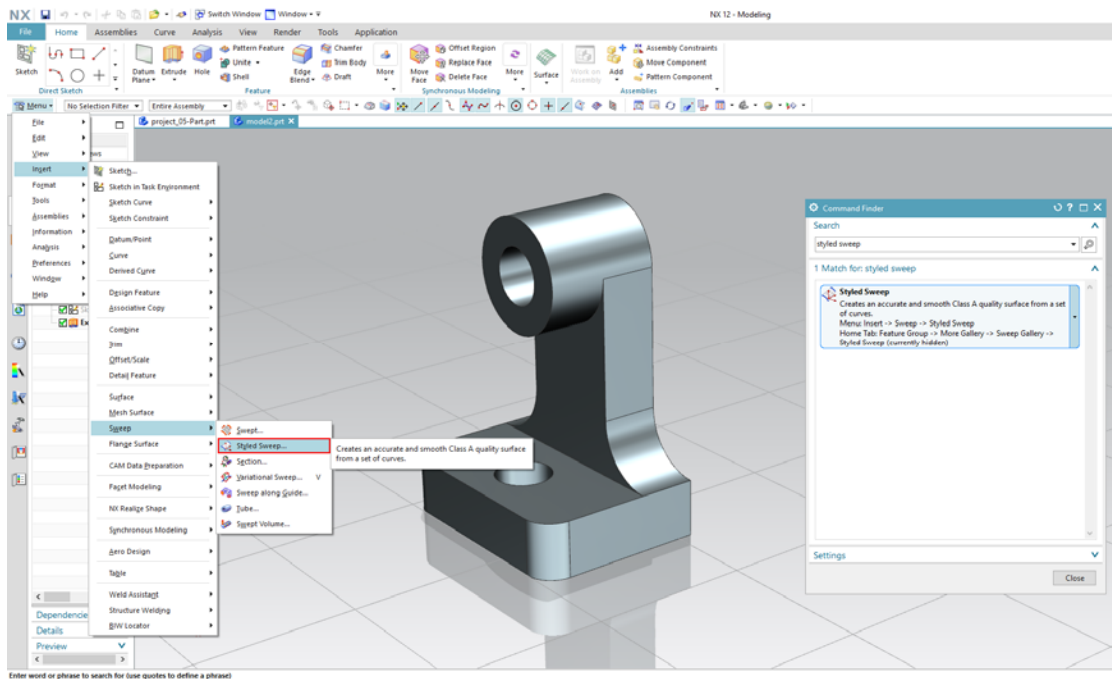
### 2.3.2.3 Command Finder

If you do not know where to find a command, use *Command Finder*. Let's say we have forgotten where the *Styled Sweep* is.

- Type **sweep** in the *Command Finder*
- Hover the mouse over **Styled Sweep**
- NX will show you the path to the command: **Menu → Insert → Sweep → Styled Sweep**

OR

- Type **sweep** in the *Command Finder*
- Click on **Styled Sweep** in the *Command Finder* window



### 2.3.2.4 Top-border

The most important button in the top-border is the *menu* button. Most of the features and functions of the software are available in the *menu*. The *Selection Bar* displays the selection options. These options include the *Filters*, *Components/Assembly*, and *Snap Points* for selecting features. Most common buttons in the *View* tab are also displayed in the *Top-border*.

### 2.3.2.5 Resource Bar

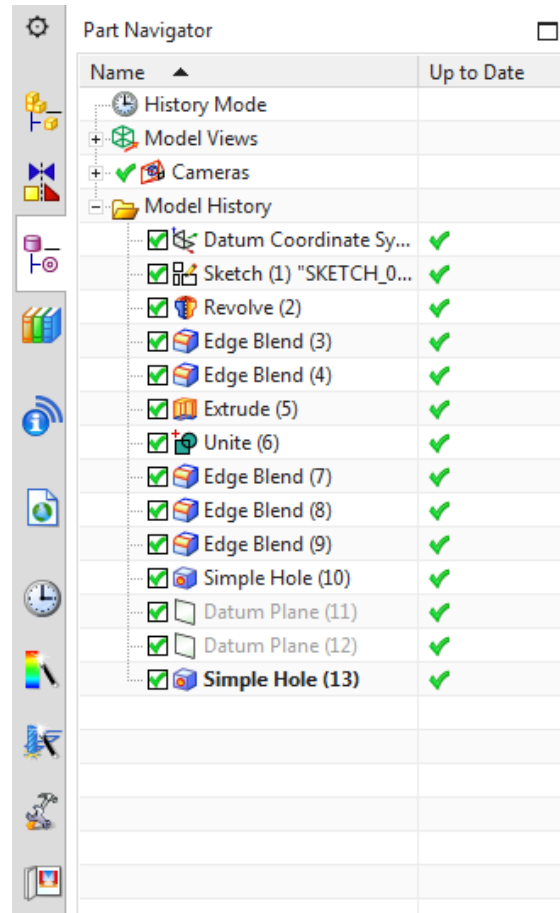
The *Resource Bar* features icons for a number of pages in one place using very little user interface space. NX 12 places all navigator windows (*Assembly*, *Constraint* and *Part*) in the *Resource Bar*, as well as the *Reuse Library*, *HD3D Tools*, *Web Browser*, *History Palette*, *Process Studio*, *Manufacturing Wizards*, *Roles* and *System Scenes*. Two of the most important widows are explained below.

## Part Navigator

- Click on the **Part Navigator** icon, the third icon from the top on the **Resource bar**

The *Part Navigator* provides a visual representation of the parent-child relationships of features in the work part in a separate window in a tree type format. It shows all the primitives, entities used during modeling. It allows you to perform various editing actions on those features. For example, you can use the *Part Navigator* to *Suppress* or *Unsuppress* the features or change their parameters or positioning dimensions. Removing the green tick mark will ‘Suppress’ the feature. The software will give a warning if the parent child relationship is broken by suppressing any particular feature.

The *Part Navigator* is available for all NX applications and not just for modeling. However, you can only perform feature-editing operations when you are in the Modeling module. Editing a feature in the *Part Navigator* will automatically update the model. Feature editing will be discussed later.



## History

- Click on the **History** icon, the seventh from the top on the **Resource bar**

The *History Palette* provides fast access to recently opened files or other palette entries. It can be used to reload parts that have been recently worked on or to repeatedly add a small set of palette items to a model.

The *History Palette* remembers the last palette options that were used and the state of the session when it was closed. NX stores the palettes that were loaded into a session and restores them in the next session. The system does not clean up the *History Palette* when parts are moved.

To re-use a part, drag and drop it from the History Palette to the Graphics Window. To reload a part, click on a saved session bookmark.

### 2.3.2.6 Cue Line

The *Cue Line* displays prompt messages that indicate the next action that needs to be taken. To the right of the Cue line, the *Status Line* is located which displays messages about the current options or the most recently completed function.

The *Progress Meter* is displayed in the *Cue Line* when the system performs a time-consuming operation such as loading a large assembly. The meter shows the percentage of the operation that has been completed. When the operation is finished, the system displays the next appropriate cue.

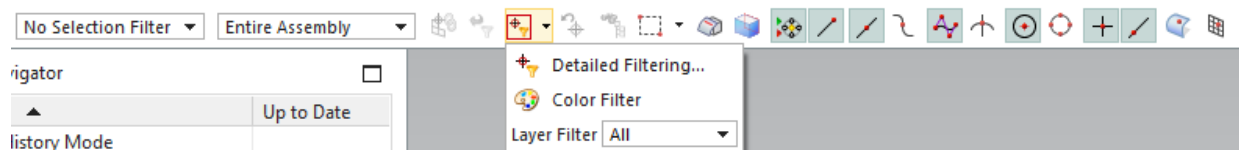
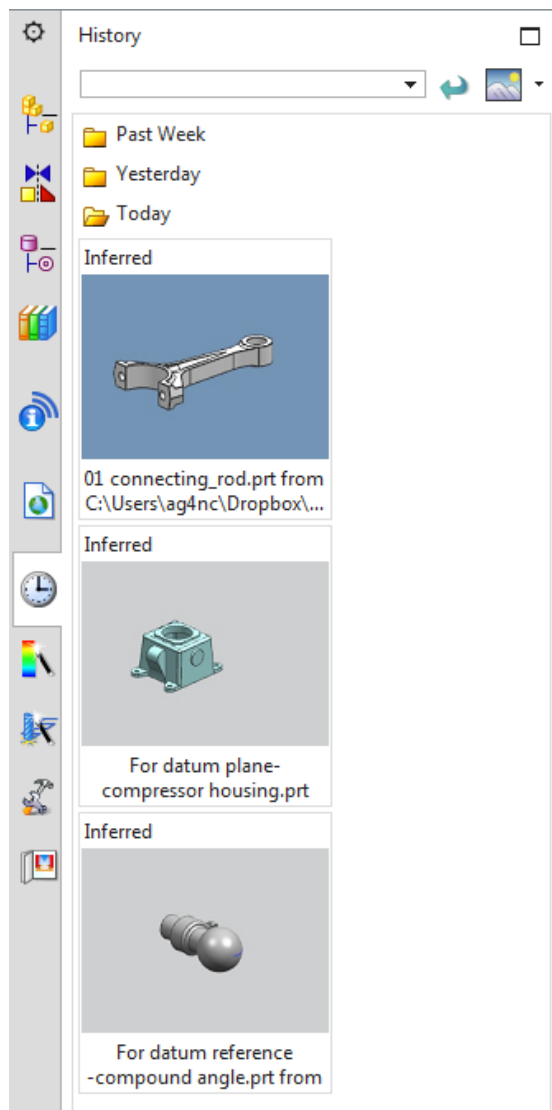
### 2.3.3 Geometry Selection

You can filter the selection method, which facilitates easy selection of the geometry in a close cluster. In addition, you can perform any of the feature operation options that NX 12 intelligently provides

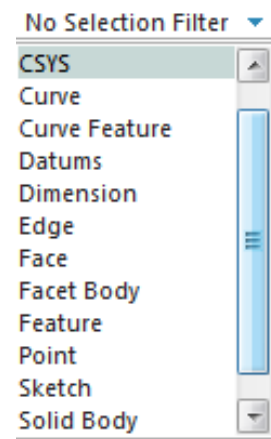
depending on the selected entity. Selection of items can be based on the degree of the entity like, selection of *Geometric* entities, *Features* and *Components*. The selection method can be opted by choosing one of the icons in the *Selection Toolbar*.

#### 2.3.3.1 Feature Selection

Clicking on any of the icons lets you select the features in the part file. It will not select the basic entities like edges, faces etc. The features selected can also be applied to a part or an entire assembly depending upon the requirement.



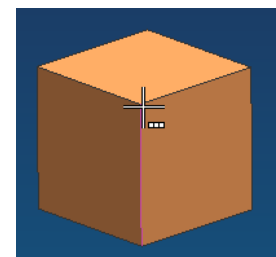
Besides that, the filtering of the features can be further narrowed down by selecting one of the desired options in the drop-down menu as shown in the figure. For example, selecting *Curve* will highlight only the curves in the screen. The default is *No Selection Filter*.



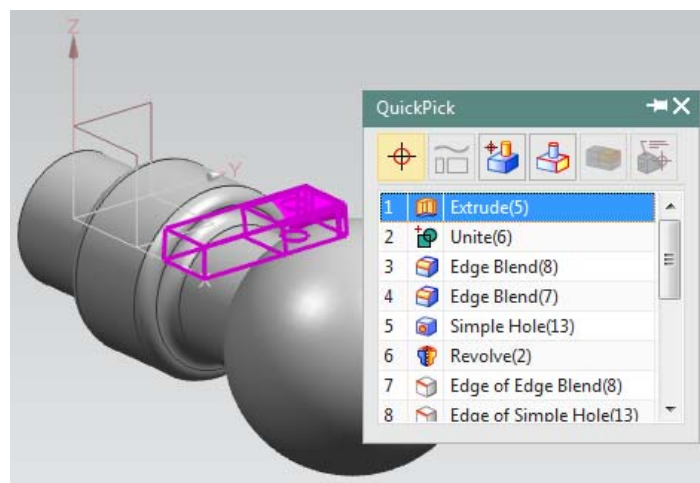
### 2.3.3.2 General Object Selection

Navigate the mouse cursor closer to the entity until it is highlighted with a magenta color and click the left mouse button to select any geometric entity, feature, or component.

If you want to select an entity that is hidden behind the displayed geometry, place the mouse cursor roughly close to that area on the screen such that the cursor ball occupies a portion of the hidden geometry projected on the screen. After a couple of seconds, the ball cursor turns into a *plus* symbol as shown in the figure. Click the left mouse button (MB1) to get a *Selection Confirmation* dialog box as shown in the



following figure below. This *QuickPick* menu consists of the list of entities captured within the ball of the cursor. The entities are arranged in ascending order of the *degree* of the entity. For example, edges and vertices are assigned lower numbers while solid faces are given higher numbers. By moving the cursor on the numbers displayed, NX 12 will highlight the corresponding entity on the screen in a magenta color.



### 2.3.4 User Preferences

- Choose **Preferences** on the **Menu button** (located to top left of the main window) to find the various options available

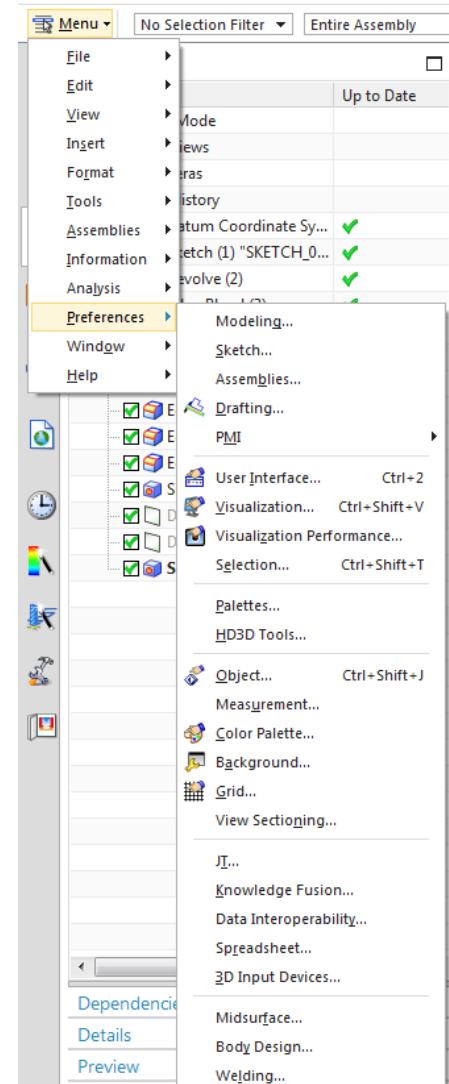


User Preferences are used to define the display parameters of new objects, names, layouts, and views. You can set the layer, color, font, and width of created objects. You can also design layouts and views, control the display of object and view names and borders, change the size of the selection ball, specify the selection rectangle method, set chaining tolerance and method, and design and activate a grid. Changes that you make using the Preferences menu override any counterpart customer defaults for the same functions.

### 2.3.4.1 User Interface

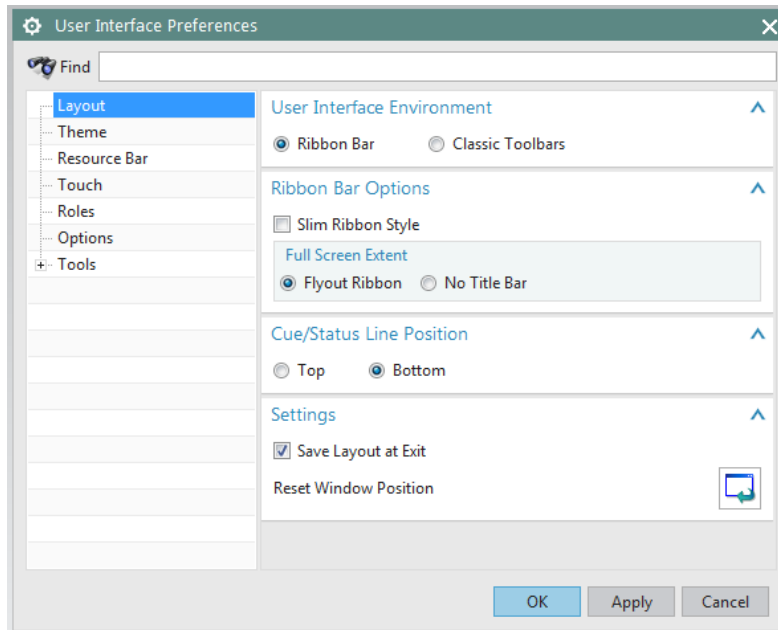
- Choose **Preferences** → **User Interface** to find the options in the dialog box

The *User Interface* option customizes how NX works and interacts to specifications you set. You can control the location, size and visibility status of the main window, graphics display, and information window. You can set the number of decimal places (precision) that the system uses for both input text fields and data displayed in the information window. You can also specify a full or small dialog for file selection. You can also set macro options and enable a confirmation dialog for *Undo* operations.



- The *Layout* tab allows you to select the *User Interface Environment*
- The *Touch* tab lets you use touch screens
- The *Options* tab allows you, among others, to set the precision level (in the *Information Window*)
- The *Journal* tab in the *Tools* allows you to use several programming languages
- The *Macro* tab in the *Tools* allows you to set the pause while displaying animation



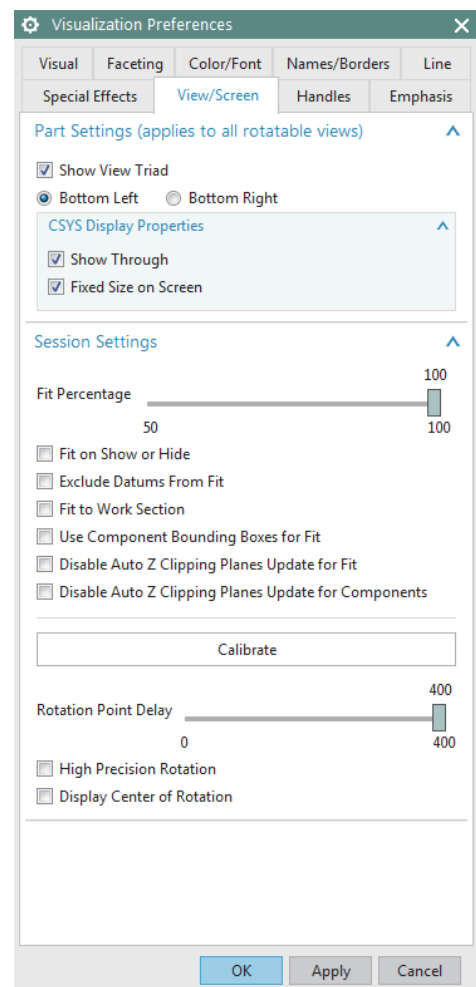


#### 2.3.4.2 Visualization

- Choose **Preferences** → **Visualization** to find the options in the dialog box

This dialog box controls attributes that affect the display in the graphics window. Some attributes are associated with the part or with particular *Views* of the part. The settings for these attributes are saved in the part file. For many of these attributes, when a new part or a view is created, the setting is initialized to the value specified in the *Customer Defaults* file. Other attributes are associated with the session and apply to all parts in the session. The settings of some of these attributes are saved from session to session in the registry. For some session attributes, the setting can be initialized to the value specified by customer default, an environment variable.

- Choose **Preferences** → **Color Pallet** to find the options in the dialog box



- Click on **Preferences** → **Background** to get another pop up Dialog box. You can change your background color whatever you want

The background color refers to the color of the background of the graphics window. NX supports graduated backgrounds for all display modes. You can select background colors for *Shaded* or *Wireframe* displays. The background can be *Plain* or *Graduated*. Valid options for all background colors are 0 to 255.

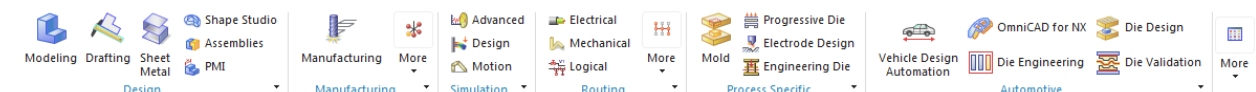
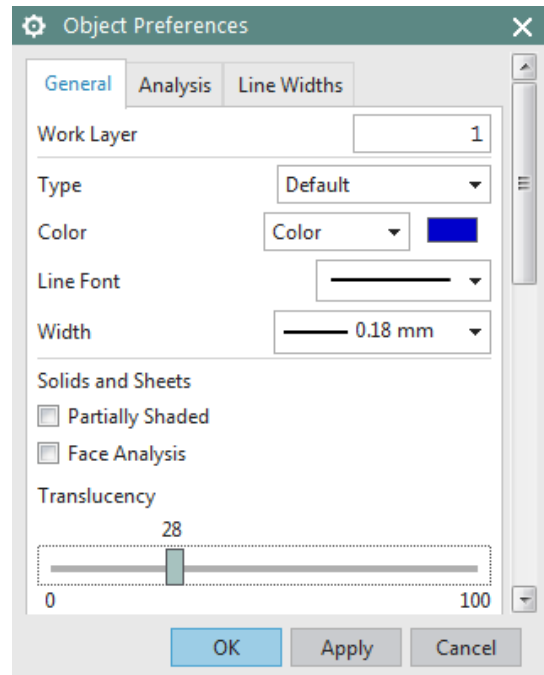
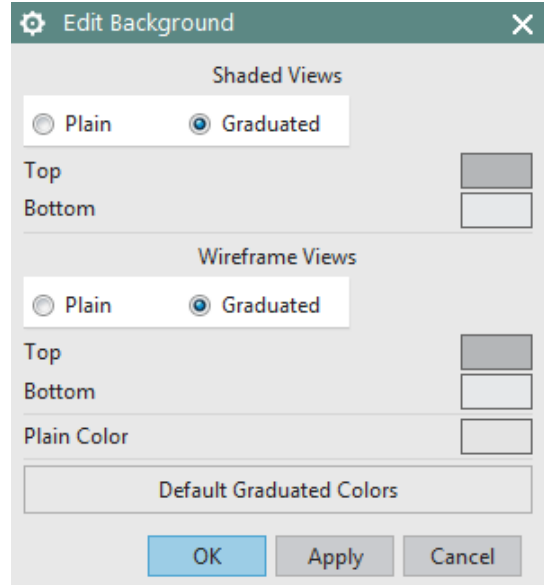
You can change and observe the *Color* and *Translucency* of objects.

- Click **Preferences** → **Object**

This will pop up a dialog window *Object Preferences*. You can also apply this setting to individual entities of the solid. For example, you can click on any particular surface of the solid and apply the *Display* settings.

### 2.3.5 Applications

*Applications* can be opened using the *File* option located at the top left corner of the main window OR the *Applications* tab above the *Ribbon bar*. You can select the type of application you want to run. For example, you can select *Modeling*, *Drafting*, *Assembly*, and so on as shown in the figure. The default *Application* that starts when you open a file or start a new file is *Modeling*. We will introduce some of these *Application* in the next chapters.



## 2.4 LAYERS

*Layers* are used to store objects in a file, and work like containers to collect the objects in a structured and consistent manner. Unlike simple visual tools like *Show* and *Hide*, *Layers* provide a permanent way to organize and manage the visibility and selectability of objects in your file.

### 2.4.1 Layer Control

With NX 12, you can control whether objects are visible or selectable by using *Layers*. A *Layer* is a system-defined attribute such as color, font, and width that all objects in NX 12 must have. There are 256 usable layers in NX 12, one of which is always the *Work Layer*. Any of the 256 layers can be assigned to one of four classifications of status.

- *Work*
- *Selectable*
- *Visible Only*
- *Invisible*

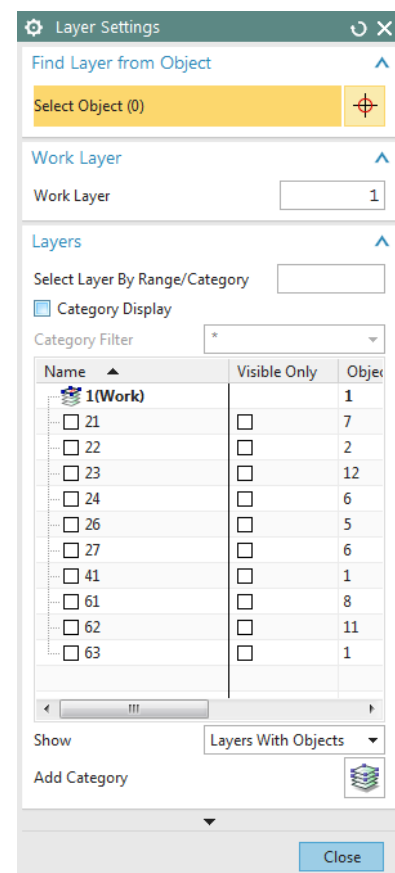
The *Work Layer* is the layer that objects are created ON and is always visible and selectable while it remains the *Work Layer*. *Layer 1* is the default *Work Layer* when starting a new part file. When the *Work Layer* is changed to another type of layer, the previous *Work Layer* automatically becomes *Selectable* and can then be assigned a status of *Visible Only* or *Invisible*.

The number of objects that can be on one layer is not limited. You have the freedom to choose whichever layer you want to create the object on and the status of that layer.

To assign a status to a layer or layers,

- Choose **View** → **Layer Settings**

However, it should be noted that the use of company standards in regards to layers would be advantageous to maintain a consistency between files.



## 2.4.2 Commands in Layers

We will follow simple steps to practice the commands in *Layers*. First, we will create two objects (Solids) by the method as follows. The details of *Solid Modeling* will be discussed in the next chapter. The solids that we draw here are only for practice in this chapter.

- Choose **File** → **New**

Name the file and choose a folder in which to save it. Make sure you select the units to be *millimeters* in the drop-down menu. Choose the file type as *Model*

- Choose **Menu** → **Insert** → **Design Feature** → **Cone**
- Choose **Diameter and Height** under **Type**
- Click **OK**
- **Right-click** on the screen and choose **Orient View** → **Trimetric**
- **Right-click** on the screen and choose **Rendering Style** → **Shaded**

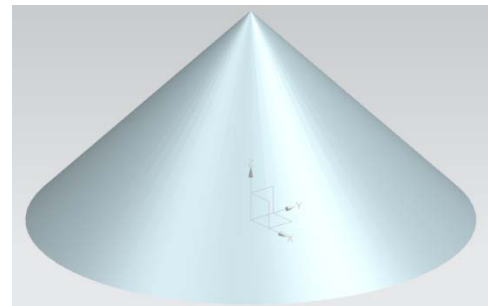
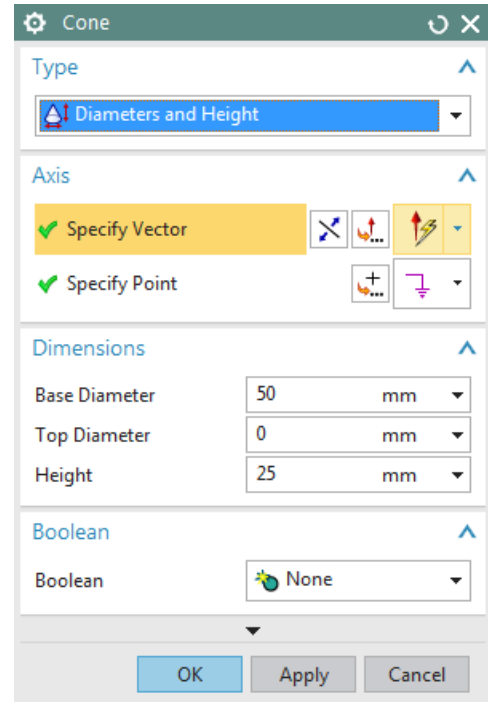
You will be able to see a solid cone similar to the picture on right.

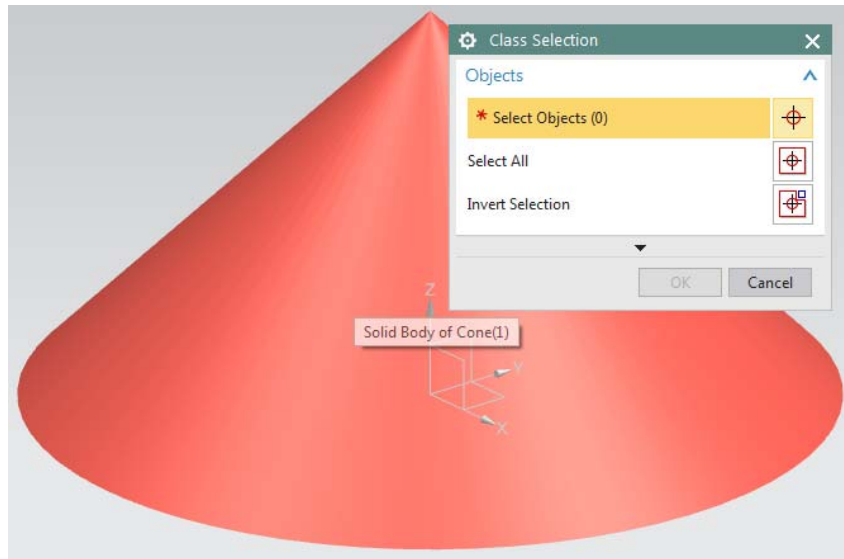
Now let us practice some *Layer* Commands.

- Choose **View** → **Move to Layer**

You will be asked to select an object

- Move the cursor on to the *Cone* and click on it so that it becomes highlighted
- Click **OK**



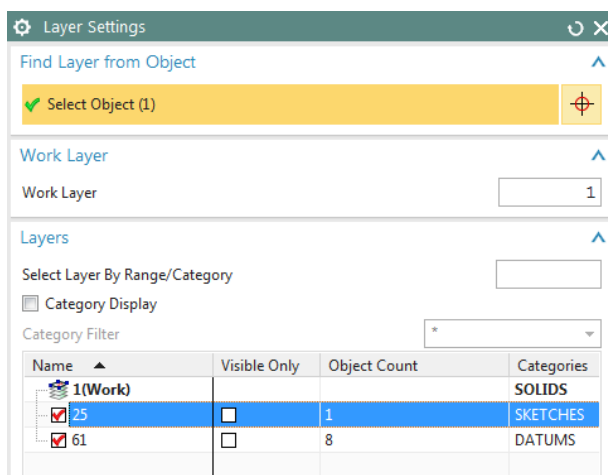
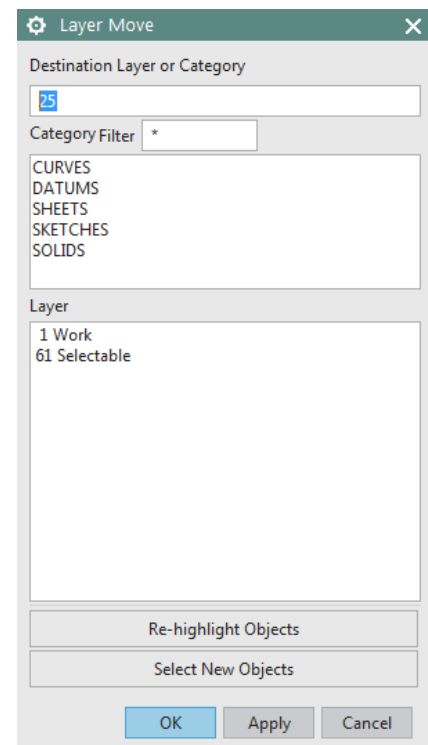


- In the **Destination Layer or Category** space at the top of the window, type 25 and Click **OK**

The *Cone* has now gone to the 25<sup>th</sup> layer. It can no longer be seen in Layer 1.

- To see the Cone, click **View → Layer Settings**
- You can see that *Layer 25* has the object whereas the default *Work Layer 1* has no objects.

The Cone will again be seen on the screen. Save the file as we will be using it later in the tutorial.

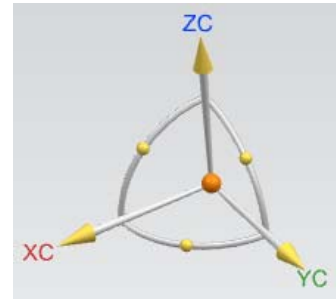


## 2.5 COORDINATE SYSTEMS

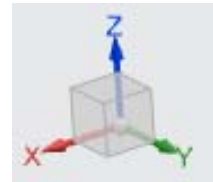
There are different coordinate systems in NX. A three-axis symbol is used to identify the coordinate system.

### 2.5.1 Absolute Coordinate System

The *Absolute Coordinate System* is the coordinate system from which all objects are referenced. This is a fixed coordinate system and the locations and orientations of every object in NX 12 modeling space are related back to this system. The *Absolute Coordinate System* (or Absolute CSYS) also provides a common frame of reference between part files. An absolute position at  $X=1$ ,  $Y=1$ , and  $Z=1$  in one part file is the same location in any other part file.



The *View Triad* on the bottom-left of the *Graphics* window is ONLY a visual indicator that represents the ORIENTATION of the *Absolute Coordinate System* of the model.



### 2.5.2 Work Coordinate System

The *Work Coordinate System* (WCS) is what you will use for construction when you want to determine orientations and angles of features. The axes of the WCS are denoted XC, YC, and ZC. (The “C” stands for “current”). It is possible to have multiple coordinate systems in a part file, but only one of them can be the work coordinate system.

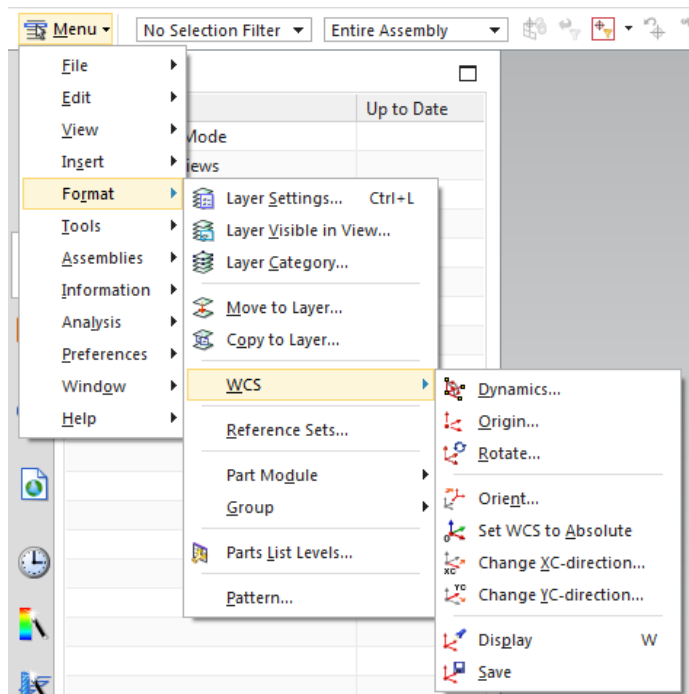
### 2.5.3 Moving the WCS

Here, you will learn how to translate and rotate the WCS.

- Choose **Menu** → **Format** → **WCS**

#### 2.5.3.1 Translate the WCS

This procedure will move the WCS origin to any point you specify, but the



orientation (direction of the axes) of the WCS will remain the same.

- Choose **Menu** → **Format** → **WCS** → **Origin**

The *Point Constructor* dialog is displayed. You either can specify a point from the drop down menu at the top of the dialog box or enter the X-Y-Z coordinates in the XC, YC, and ZC fields.

The majority of the work will be in relation to the *Work Coordinate System* rather than the *Absolute Coordinate System*. The default is the WCS.

### 2.5.3.2 Rotate the WCS

You can also rotate the WCS around one of its axes.

- Choose **Menu** → **Format** → **WCS** → **Rotate**

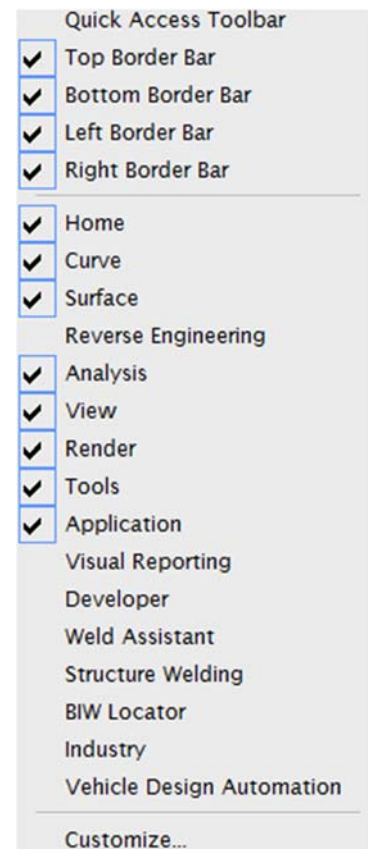
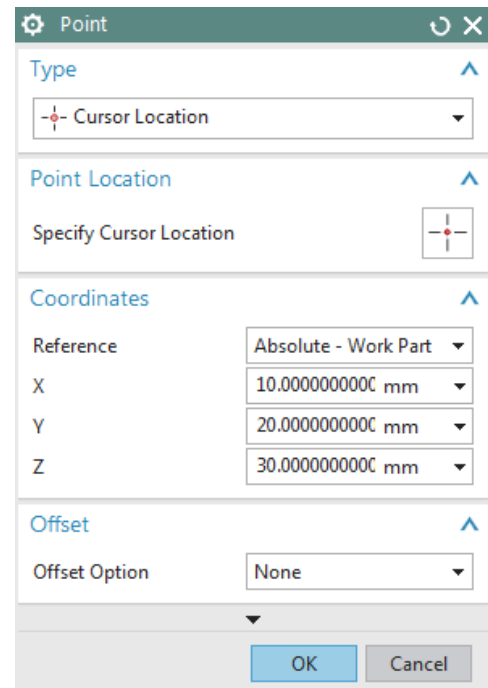
The dialog shows six different ways to rotate the WCS around an axis. These rotation procedures follow the right-hand rule of rotation. You can also specify the angle to which the WCS be rotated.

You can save the current location and orientation of the WCS to use as a permanent coordinate system.

- Choose **Menu** → **Format** → **WCS** → **Save**

## 2.6 TOOLBARS

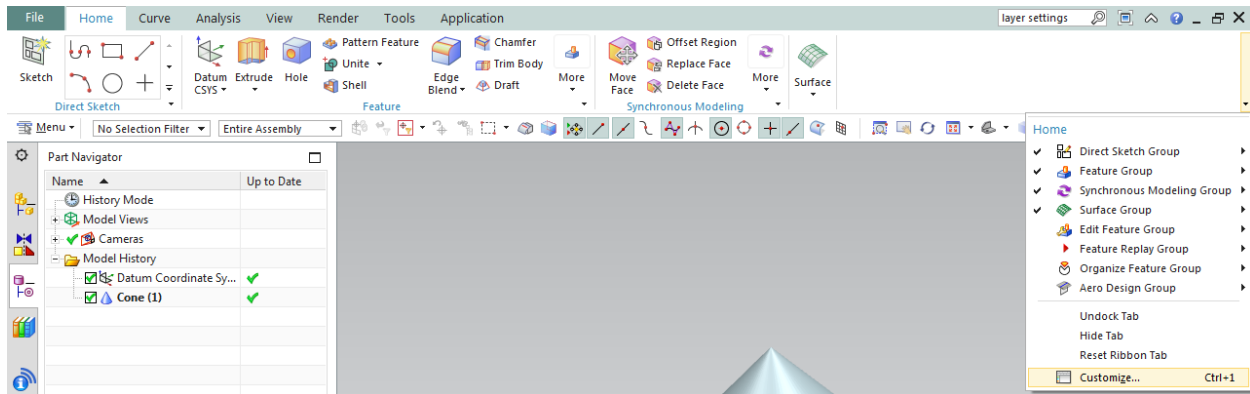
Toolbars contain icons, which serve as shortcuts for many functions. The figure on the right shows the main Toolbar items normally displayed. However, you can find many more icons for different feature commands, based on the module selected and how the module is customized.



- **Right-Clicking** anywhere on the existing toolbars gives a list of other **Toolbars**. You can add any of the toolbars by checking them.

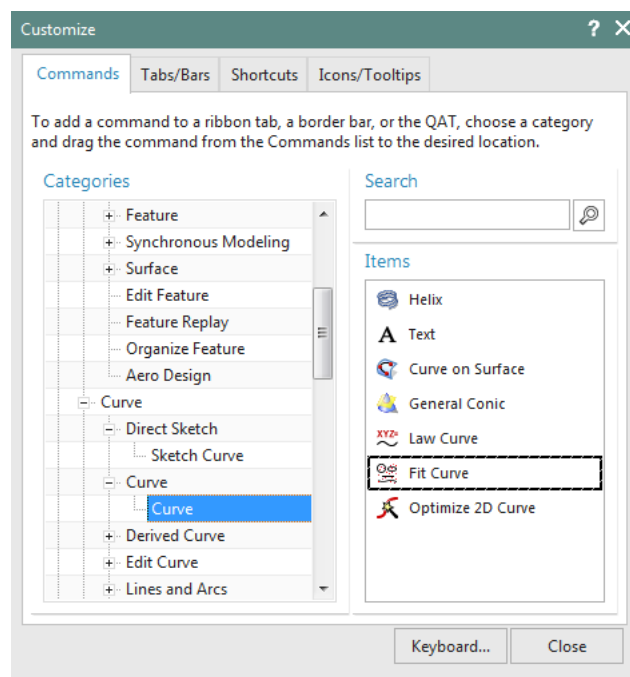
Normally, the default setting should be sufficient for most operations but during certain operations, you might need additional toolbars. If you want to add buttons pertaining to the commands and toolbars,

- Click on the **pull-down arrow** on any of the Toolbars and choose **Customize**.



This will pop up a *Customize* dialog window with all the Toolbars and commands pertaining to each Toolbar under *Commands* tab. To add a command,

- Choose a category and drag the command from the *Commands list* to the desired location.





You can customize the settings of your NX 12 interface by clicking on the *Roles* tab on the *Resource Bar*.

The *Roles* tab has different settings of the toolbar menus that are displayed on the NX 12 interface. It allows you to customize the toolbars you desire to be displayed in the Interface.

